10.2478/sggw-2018-0003

Annals of Warsaw University of Life Sciences – SGGW
Land Reclamation No 50 (1), 2018: 33–42
(Ann. Warsaw Univ. of Life Sci. – SGGW, Land Reclam. 50 (1), 2018)

The length of the hydraulic jump on the basis of physical and numerical modeling

JANUSZ URBAŃSKI, PIOTR SIWICKI, ADAM KICZKO, ADAM KOZIOŁ, MARCIN KRUKOWSKI
Faculty of Civil and Environmental Engineering, Warsaw University of Life Sciences – SGGW

Abstract: *The length of the hydraulic jump on the basis of physical and numerical modeling.* The outcomes of physical and numerical modeling of the sluice gate outflow are presented. The measured velocity distributions in verticals of a physical model were compared with results of numerical modeling, obtained using ANSYS Fluent software. The research goal was verification of suitability of the computational fluid dynamic (CFD) approach in determination of the hydraulic jump length at the outflow of the flow control structure. Studies were performed for the model of the sluice gate and stilling basin with two setups of baffle blocks: in one and two rows. The jump lengths were estimated by an analysis of vertical velocity profiles at the outflow. Two rows of baffle blocks in the stilling basin allowed to reduce the length of the hydraulic jump by 5–10%, comparing to the length with the single row of blocks. The computational fluid dynamic approach underestimated the length of the hydraulic jump by 4–7%, comparing to the physical model.

Key words: sluice gate, hydraulic jump, physical modeling, numerical modeling, CFD

INTRODUCTION

Many studies were devoted to the problem of a hydraulic jump being a transition from the supercritical (rapid) to subcritical (tranquil) flow, because of its practical importance. In civil engineering the hydraulic jump is used to dissipate stream energy. It is usually, so called, the submerged hydraulic jump, maintained in the stilling basin downstream the flow control device. A design of such outflow structures accounts for conjugate depths: first (h_1) and second (h_2) and a length of the hydraulic jump (L_0).

The length of the hydraulic jump (L_0) is measured starting from a cross section of the first conjugate depth (h_1). However, the ending cross section is difficult to identify. It is because of high turbulence intensity in the hydraulic jump and immediately downstream it, resulting in continuous changes in properties of the water stream (Wu and Rajaratnam 1996, Habibzadeh et al. 2016, Kozioł et al. 2017). Hager et al. (1990) proposed classifying the cross section with stagnating water at the surface as the end of the hydraulic jump. So called the length of the water roller is measured between the first conjugate depth and such a cross section. Such a definition seems to be the most appropriate, as by this way the length can be easily and objectively determined.

There are many empirical formulae for the length of the hydraulic jump (Čertousov 1962, Hager et al. 1990, Bessaih and Rezak 2002, Kisiel 2005, Gupta et al. 2010). These formulae are usually based on simple relationships involving conjugate depths h_1 i h_2, Froude number Fr_1 in the cross section of the first conjugate depth and various empirical coefficients. Urbański (2008) provided a review of chosen methods of the water roller length (L_0). Formulae apply mostly for the unsubmerged hydraulic jump appearing in a rectangular channel with flat bottom. Less often the submerged hydraulic jump was considered, which properties are being shaped by energy dissipation devices at the outflow. Submergence affects the flow structure of the jump and reduces it length (Urbański 2008). The effect of baffle blocks at the outflow on the length of the hydraulic jump is still poorly recognized. Such baffle blocks, made of concrete of appropriate shapes with incline face surfaces allow reducing the hydraulic jump length by almost 48%. Urbański (2015) showed for the submerged jump in the stilling basin that it is possible to reduce the length of the jump by 7–8% with a single row of baffle blocks and by 15–19% with two rows.

Polish Regulation of the Minister of the Environment of 20 April 2007 on technical conditions are to be met by hydrotechnical constructions and their location in the paragraph 78, point 2 specifies that for the flow control structures of I and II class it is necessary to perform modeling of a discharge characteristic of outflow and energy dissipating devices. Construction of physical models for all design variants is laborious and expensive. As an alternative, the numerical modeling can be considered, because of its flexibility allows to obtain the result in shorter time with lower costs. Experimental data is still required for verification of numerical models and evaluation of their suitability in a certain problem.

The present study provides results of physical and numerical modeling of the flow at the weir outflow with a stilling basing and baffle blocks. Experimental studies in a hydraulic laboratory allowed to obtain vertical profiles of the stream velocity at the outflow. The observations were compared with the output of the numerical modeling, performed using computational fluid dynamic (CFD) solver ANSYS Fluent. The length of the hydraulic jump was assessed using obtained velocity distributions. The goal of the research was an evaluation of a suitability of the CFD modeling in an estimation of the length of the hydraulic jump downstream the sluice gate, in a stilling basin with baffle blocks.

METHODS

Physical model

Research was performed for the physical model of a sluice gate (Fig. 1a) in a rectangular flume, wide for B = 1.0 m. The opening height of the gate (a) was adjusted during each experiment. The submerged hydraulic jump was maintained in the stilling basin with a different setup of baffle blocks: in a single row (Fig. 1b) and two rows (Fig. 1c). During each experiment the water head upstream, the gate (H) and the outflow depth (h) were kept constant. Hydraulic properties of the stream for three analyzed flow rates (q) are given in Table 1.

Baffle blocks of shapes shown in Fig. 1a were determined according to guidelines provided by Peterka (1978), where the block height (h_s) is conditioned on Froude number $Fr_1 = \frac{q}{h_1\sqrt{gh_1}}$

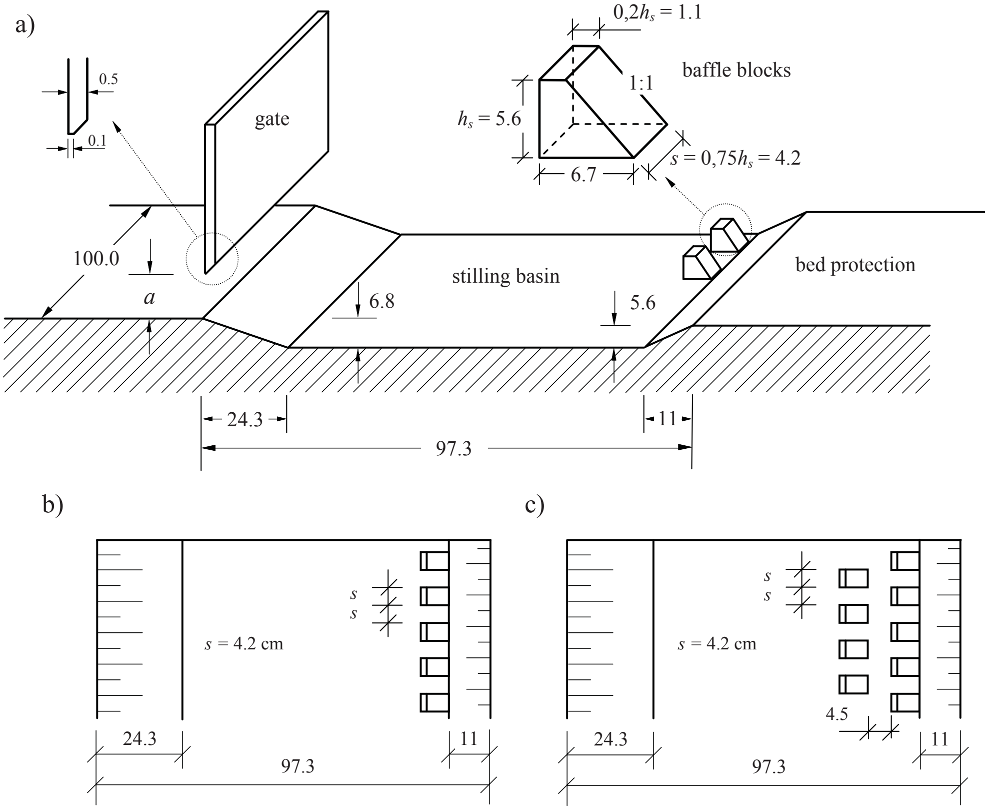

FIGURE 1. Model of the sluice gate: a – schema and base dimensions (cm); b – stilling basin with baffle block in one row; c – stilling basin with baffle blocks in two rows

TABLE 1. Hydraulic properties of flow in the model

q	a	H	h	h_1	h_2	Fr_1	σ_z	L_s
$m^2 \cdot s^{-1}$	m	m	m	m	m	–	–	m
0.049	0.032	0.417	0.133	0.020	0.147	5.7	1.28	0.08
0.073	0.049	0.445	0.165	0.030	0.176	4.4	1.27	0.12
0.097	0.064	0.462	0.193	0.039	0.200	3.9	1.24	0.16

in a cross section of the first conjugate depth (h_1). At the design flow for sizing and spacing of blocks, the lowest analyzed flow rate of $q = 0.049\ m^3 \cdot s^{-1}$ ($Fr_1 = 5.7$) was used. The first conjugate depth was found on the basis of a minimum depth of the contracted stream downstream the sluice gate and calculated as $h_1 = \varepsilon a$, where ε denotes the contraction coefficient. According to Żukowski (after Kisiel et al. 2003), the contraction coefficient depends on the ratio a to H.

The second conjugate depth was calculated using:

$$h_2 = \frac{h_1}{2}\left(\sqrt{1+8\frac{q^2}{gh_1^3}}-1\right).$$

It was assumed that the hydraulic jump ending section is the flow stagnation point, with the zero longitudinal velocity at the surface. The position of this section was found on the basis of measured velocities at the outflow (Urbański 2008). The measurement verticals were localized in an axial plane of the flume along the stilling basin and horizontal bottom at the outflow (Fig. 2). Measurement points were arranged as follows: p_1 1 cm above the bottom, p_2 $0.2h_w$ above the bottom, $p_3 - 0.5h_w$, $p_4 - 0.7h_w$, and p_5 2 cm bellow a water surface. In verticals localized in baffle blocks sections, the velocity was measured in three points. The stream depth (h_w) at the outflow was varying and measured in each vertical using a pin gauge.

The velocity was measured using an electromagnetic sensor PEMS with sampling interval of 0.1 s. Instantaneous velocities in each point were measured for 120 s, resulting in time series consisting of 1,200 elements. Values were averaged, obtaining mean values v_m, which were used to produce a vertical velocity distributions. Velocity profiles allowed to identify the velocity stagnation line (Bogomolov and Michalov 1972) near the hydraulic jump and by the extrapolation also the stagnation point at the surface. This point was used as the position of the ending section of the hydraulic jump.

The length of the hydraulic jump was determined as the difference between the distance from the sluice gate to the stagnation point (L_w) and the length of the supercritical stream downstream the gate, reduced by the gate thickness $z = 5$ mm: $L_0 = L_w - L'_s$, where $L'_s = L_s - z$ (Fig. 2). The $L_s = 2.5a$ was determined according to the "Guidelines the design of water drainage systems – weirs" (1970) for the sluice gate flow without a weir. The submergence ratio of the hydraulic jump (σ_z) was calculated using the following formula (Dąbkowski et al. 1982): $\sigma_z = \frac{h + d + \Delta z}{h_2}$, where d stands for the depth of the stilling basin, h for stream depth at the outflow. A height (Δz) denotes the water level raise at the stilling basin outflow and can be determined

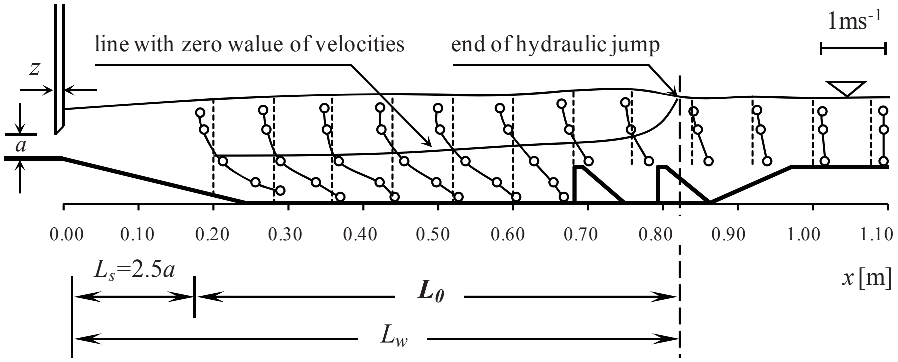

FIGURE 2. Identifying the localization of the hydraulic jump end, according to the zero velocity line

as: $\Delta z = \frac{v_h^2 - v_2^2}{2g}$, where v_h and v_2 stream velocity in the cross section of depth h and h_2, respectively. Obtained values are given in Table 1.

Numerical model

The numerical model was performed on the basis of the physical model geometry, using the CFD software of ANSYS Fluent. The open channel flow was described using the volume of fluid approach with the Reynolds-averaged Navier–Stokes equations (Kiczko et al. 2015). After analyzing five different turbulent models available in ANSYS Fluent (Fluent 2001), the k-epsilon closure was chosen, as the most suitable for the analyzed flow (Urbański and Siwicki 2007). Calculations were performed for unsteady flow conditions (transident). Figure 3a presents schema of the numerical model developed using ANSYS Fluent with the obtained water surface for one of the experimental variants. The further analyses were performed for the axial plane (Fig. 3b) and velocity verticals corresponding to the laboratory measurements.

RESULTS AND DISCUSSION

The computed velocity distributions in verticals downstream the gate using CFD model were compared with those obtained in laboratory experiments. This allowed to verify the suitability of the numerical approach (Figs. 4, 5 and 6).

The obtained velocity verticals using both modeling approaches are typical for the water stream downstream the structure with a sluice gate. The highest values were observed near the bottom, that results from the bottom transit stream. It should be noted that the numerical model underestimated the maximum values comparing to the physical one. These differences might be the result of an improper representation of the bottom roughness in the CFD model. Negative values of the velocity near the surface were reported by both models. For each model on the basis of the vertical velocity profiles, lines of the zero velocity were determined. In Figures 4, 5 and 6 these lines are presented as solid lines for the experimental data (L_{wLab}) and dashed ones for the CFD model (L_{wCFD}). The ending sections of the hydraulic jump were obtained by extrapolation of zero velocity lines towards the water surface.

a) b)

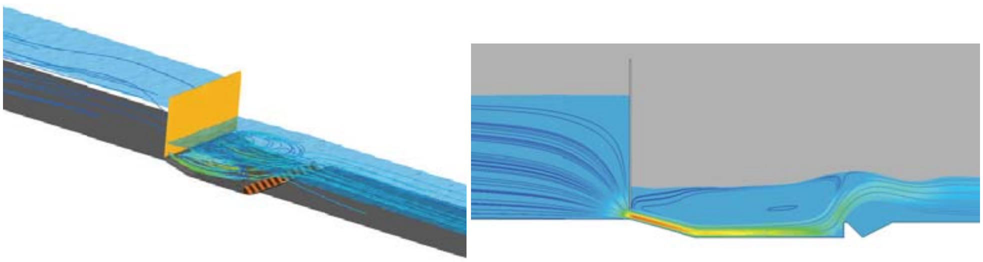

FIGURE 3. Visualization of the CFD ANSYS model output: a – water surface, b – velocity izolines in the flume axial plane

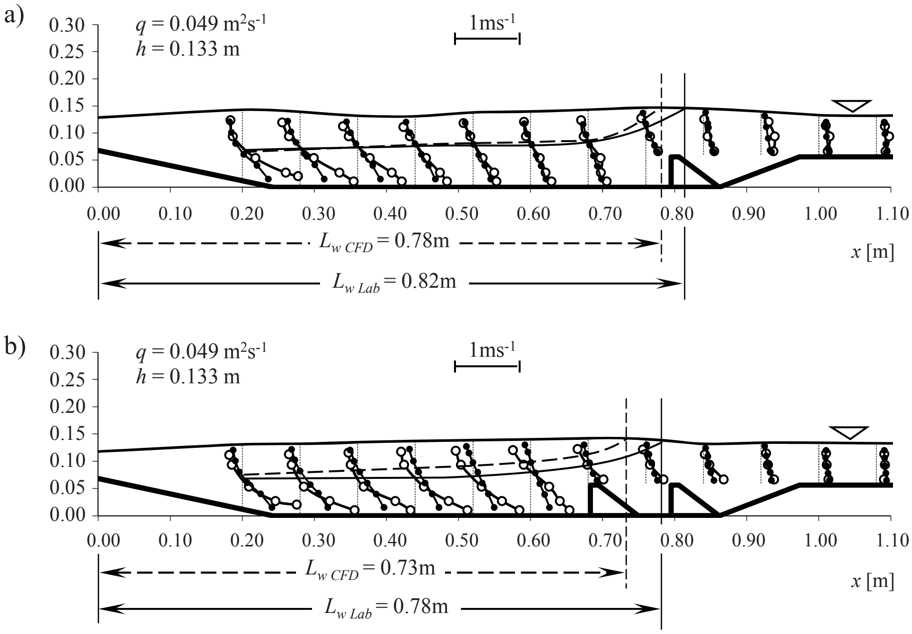

FIGURE 4. Measured and computed velocity distributions using the CFD approach for the model with baffle blocks in (a) single row, (b) two rows for the flow rate of $q = 0.049\ \mathrm{m^2 \cdot s^{-1}}$

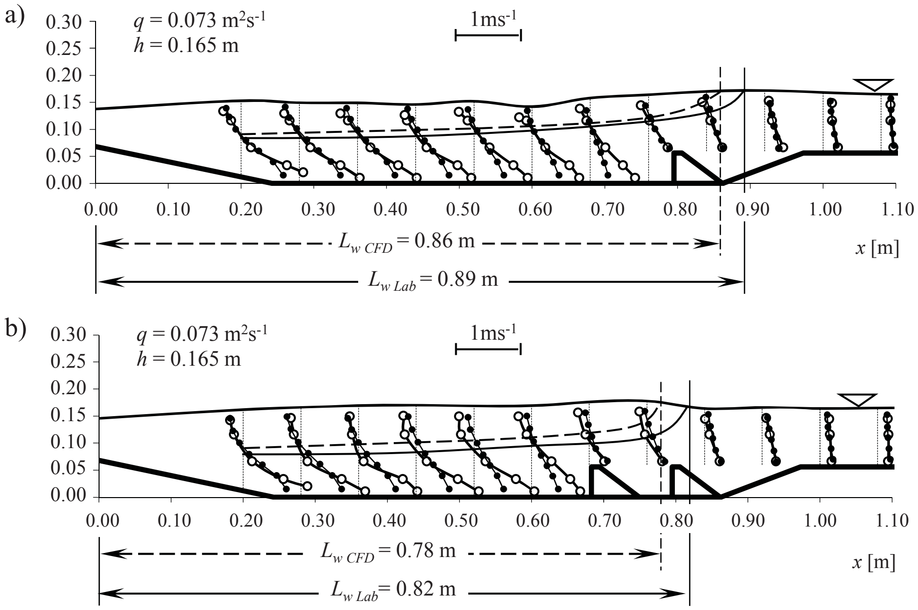

FIGURE 5. Measured and computed velocity distributions using the CFD approach for the model with baffle blocks in (a) single row, (b) two rows for the flow rate of $q = 0.073\ \mathrm{m^2 \cdot s^{-1}}$

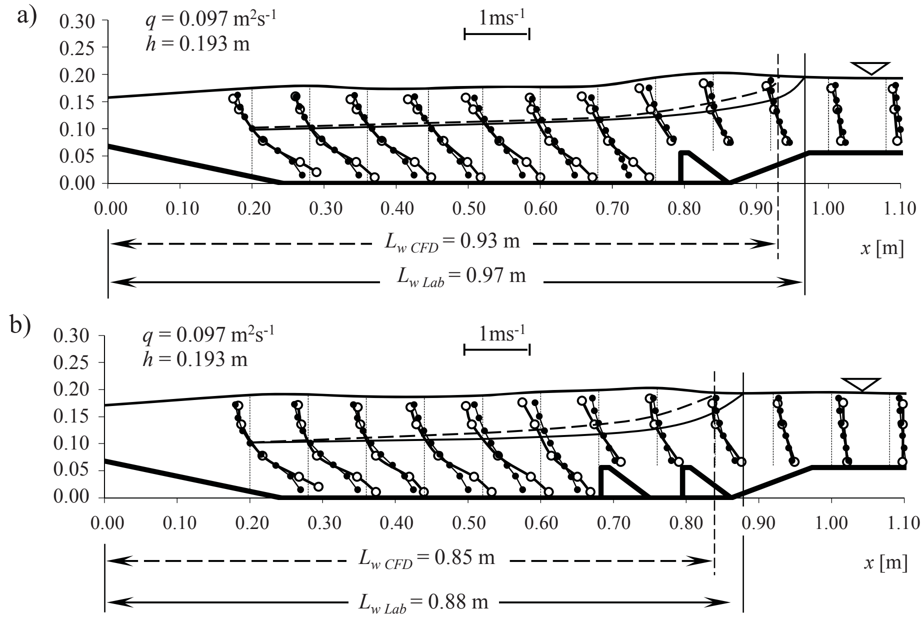

FIGURE 6. Measured and computed velocity distributions using the CFD approach for the model with baffle blocks in (a) single row, (b) two rows for the flow rate of $q = 0.097\ m^2 \cdot s^{-1}$.

TABLE 2. Hydraulic jump lengths calculated using physical and numerical models

q	Energy dissipation setup	Model	L_w	L_s	L_0	$\frac{L_{0(CFD)}}{L_{0(Lab)}}$
$m^2 \cdot s^{-1}$			m	m	m	–
0.049	Sz1	Lab	0.82	0.08	0.74	0.95
		CFD	0.78		0.70	
	Sz2	Lab	0.78		0.70	0.93
		CFD	0.73		0.65	
0.073	Sz1	Lab	0.89	0.12	0.77	0.96
		CFD	0.86		0.74	
	Sz2	Lab	0.82		0.70	0.94
		CFD	0.78		0.66	
0.097	Sz1	Lab	0.97	0.16	0.81	0.95
		CFD	0.93		0.77	
	Sz2	Lab	0.88		0.72	0.96
		CFD	0.85		0.69	

Table 2 provides the lengths of the hydraulic jump determined using physical and numerical modeling. The setup of energy dissipation devices is marked as follows: Sz1 – stilling basin with baffle blocks in the single row, Sz2 – stilling basin with baffle blocks in two rows. The results indicate that two rows of baffles blocks in the stilling basin reduce the length of the hydraulic jump by 5–10% comparing with the single row setup of baffle blocks. The term $\frac{L_{0(CFD)}}{L_{0(Lab)}}$ in Table 2 is a ratio of the jump length computed using the CFD approach to the length obtained on the basis of the experimental data. The ratio values were within the range of 0.93–0.96, denoting that hydraulic jump lengths obtained using the CFD approach were underestimated by 4–7%.

CONCLUSIONS

The analysis of results of physical and numerical modeling, which goal was to verify the suitability of the CFD approach in determining the hydraulic jump lengths in the stilling basin with baffle blocks, allowed to draw the following conclusions:

1. The length of the hydraulic jump at the outflow of the control structure is difficult to determine, because of complex structure of the stream, affected by intensive turbulence. Identification of the beginning and ending sections of the hydraulic jump is a difficult task.
2. Two rows of baffle blocks in the stilling basin allowed to reduce the length of the hydraulic jump by 5–10%, comparing to the length with the single row of blocks. The CFD approach underestimated the length of the hydraulic jump by 4–7%, comparing to the physical model.
3. The application of the numerical CFD model to analyze the kinematic structure of the stream at the outflow of the flow control structure is justified. However, the obtained differences in numerical and physical models for two setups of energy dissipation devices suggests that the final design of the construction should be verified on the basis of physical modeling.

REFERENCES

BESSAIH N., REZAK A. 2002: Effect of baffle blocks with a sloping front face on the length of the jump. J. Civil Eng. 30 (2): 101–108.

BOGOMOLOV A.I., MICHAJLOV K.A. 1972: Gidravlika (Hydraulics). Strojizdat, Moskva.

ČERTOUSOV M.D. 1962: Gidravlika. Specialnyj kurs. Gosenergoizdat, Moskva – Leningrad.

DĄBKOWSKI Sz.L., SKIBIŃSKI J., ŻBIKOWSKI A. 1982: Hydrauliczne podstawy projektów wodnomelioracyjnych. PWRiL, Warszawa.

Fluent 6.0 User' Guide. Fluent Inc. 2001.

GUPTA S.K., MEHTA R.C., DWIVEDI V.K. 2010: Modelling of Relative Length and Relative Energy Loss of Hydraulic Jump in Sloping Prismatic Channels for Environmental Hazards Control. 2nd International Conference on Climate Change & Sustainable Management of Natural Resources, CP–77.

HABIBZADEH A., LOEWEN M.R., RAJARATNAM N. 2016: Turbulence Measurements in Submerged Hydraulic Jumps with Baffle Blocks. Can. J. Civil Eng. 43 (6): 553–561.

HAGER W.H., BREMEN R., KAWAGOSHI N. 1990: Classical hydraulic jump: length of roller. J. Hydr. Res. 28 (5): 591–608.

KICZKO A., KUBRAK J., KUBRAK E. 2015: Experimental and numerical investigation of non-submerged flow under a sluice gate. Ann. Warsaw Univ. Life Sci. – SGGW, Land Reclam. 47 (3): 187–201.

KISIEL A. 2005. Hydrauliczne podstawy wymiarowania typowych wypadów budowli hydrotechnicznych. Wydawnictwo Politechniki Częstochowskiej, Częstochowa.

KISIEL A., MROWIEC M., MALMUR R., KISIEL J. 2003: Poradnik hydromechanika i hydrotechnika. Wydawnictwo Politechniki Częstochowskiej, Częstochowa.

KOZIOŁ A., URBAŃSKI J., KICZKO A., KRUKOWSKI M., SIWICKI P., KALENIK M. 2017: Turbulence intensity and spatial scales of turbulence after hydraulic jump over scour hole in rectangular channel. J. Hydrol. Hydromech. 65 (4): 385–394.

PETERKA A.J. 1978: Hydraulic Design of Stilling Basins and Energy Dissipators. United States Department of the Interior, Bureau of Reclamation.

Rozporządzenie Ministra Środowiska z dnia 20 kwietnia 2007 r. w sprawie warunków technicznych, jakim powinny odpowiadać budowle hydrotechniczne i ich usytuowanie. Dz.U. 2007 nr 86 poz. 579.

URBAŃSKI J. 2008: Długość odskoku hydraulicznego na modelu jazu [Length of hydraulic jump on model of the dam]. Prz. Nauk. Inż. Kształt. Środ. 1 (39): 3–12.

URBAŃSKI J. 2015: Eksperymentalne badania wpływu szykan na długość odskoku hydraulicznego [Experimental investigations of the impact of baffle blocks on the length of hydraulic jump]. Acta Sci. Pol. Formatio Circumiectus 14 (1): 189–199.

URBAŃSKI J., SIWICKI P. 2007: Zastosowanie programu CFD Fluent do obliczeń charakterystyk turbulencji strumienia w dolnym stanowisku jazu [Application of CFD Fluent program to calculations of turbulance characterictcs in downstream of weir]. Infrastruktura i Ekologia Terenów Wiejskich 4 (2): 201–209.

WU S., RAJARATNAM N. 1996: Transition from hydraulic jump to open channel flow. J. Hydraul. Eng. 122 (9): 526–528.

Wytyczne instruktażowe projektowania budowli wodno-melioracyjnych – jazy [Guidelines the design of water drainage systems – weirs]. CBSiPWM, Warszawa 1970.

Streszczenie: *Długość odskoku hydraulicznego na fizycznym i numerycznym modelu jazu.* W pracy przedstawiono wyniki fizycznego i numerycznego modelowania przepływu turbulentnego na wypadzie jazu, które wykorzystano do wyznaczenia długości odskoku hydraulicznego. W doświadczeniach na modelu laboratoryjnym wykonano pomiary prędkości strumienia w pionach na wypadzie. Uzyskane wyniki pomiarów rozkładów prędkości porównano z wynikami obliczeń numerycznych z zastosowaniem programu CFD Fluent. Celem badań była ocena możliwości zastosowania modelu numerycznego CFD do wyznaczenia długości odskoku hydraulicznego na wypadzie jazu. Badania przeprowadzono na modelu z wypływem strumienia spod zasuwy i niecką wypadową, w której dodatkowo zastosowano szykany ustawione w jednym rzędzie i w dwóch rzędach. Długość odskoku wyznaczano na podstawie analizy zmienności rozkładów prędkości strumienia na wypadzie uzyskanych na fizycznym i numerycznym modelu. W obu przypadkach wyznaczano położenie tzw. linii zerowej prędkości, łączącej punkty w obszarze odskoku hydraulicznego, w których średnia wartość podłużnej składowej prędkości była zerowa. Jako koniec odskoku hydraulicznego przyjmowano punkt na powierzchni zwierciadła wody, w którym wartość prędkości była zerowa, a lokalizację tego punktu określano w wyniku ekstrapolacji linii zerowej prędkości w kierunku zwierciadła wody. Dokonano zestawienia oraz porównania wyników pomiarów długości odskoku hydraulicznego, uzyskanych na fizycznym i numerycznym modelu z uwzględnieniem konstrukcji zastosowanych urządzeń do rozpraszania energii w kolejnych doświadczeniach, tzn. niecki z szykanami w jednym rzędzie oraz niecki z szykanami w dwóch rzędach.

Długość odskoku hydraulicznego, tworzącego się na wypadzie budowli piętrzącej, jest parametrem trudnym do ustalenia z uwagi na złożoną strukturę strumienia spowodowaną wysokim stopniem jego turbulencji. Uzyskane wyniki i ich analiza wykazały, że szykany zastosowane w niecce wypadowej w dwóch rzędach spowodowały zmniejszenie długości odskoku hydraulicznego o 5–10% w porównaniu z tą samą długością uzyskaną na modelu z szykanami ustawionymi w jednym rzędzie. Długość odskoku uzyskana na modelu CFD była mniejsza od pomierzonej na modelu fizycznym o 4–7%. Zastosowanie modeli numerycznych CFD do rozpoznania kinematycznej struktury przepływu na wypadzie budowli jest uzasadnione, szczególnie w analizach porównawczych różnych wariantów konstrukcji urządzeń do rozpraszania energii.

Słowa kluczowe: jaz, odskok hydrauliczny, modelowanie fizyczne, modelowanie numeryczne CFD

MS received 01.03.2018
MS accepted 28.03.2018

Authors' address:
Janusz Urbański
Katedra Inżynierii Wodnej
Wydział Budownictwa i Inżynierii Środowiska
ul. Nowoursynowska 166
02-787 Warszawa
Poland
e-mail: janusz_urbanski@sggw.pl